International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 4, Issue 2, Apr 2014, 35-42

OPTIMIZATION FOR INTAKE PORT

LAXMIKANT P. NARKHEDE & ATUL PATIL

Department of Mechanical Engineering, Godavari College of Engineering, Jalgaon, Maharashtra, India

ABSTRACT

The present work deals with the study of the flow within the intake port in both steady and unsteady states and analyze the results to evaluate and improve the ability of the intake port to convey air identically to all cylinders with the least possible pressure losses. Also the effect of engine speed on the volumetric efficiency has been analyzed by 2D CFD model at different engine speeds.

Optimizing airflow performance during intake port process is the main purpose for this project. Here we are going to Analyze in CFD simulation and experimental using academic engine flow model. This analysis could be used to increase efficiency of volumetric flow rate and maximizing usage of air fuel in combustion process, which reduce emission to environment. Even though air flow have been optimized on its intake port, but still intake system could be improve by considering other parts of engine also such as intake manifold, Valve etc..

The present work is related to, two important common fluid flow patterns from computational fluid dynamics (CFD) simulations, namely, effect of steady state and unsteady state analysis and effect on air motion on turbulence inside the cylinder.

The performance of the engine can be improved by efficient design of intake ports. In the process of optimizing the port flow for improving engine performance, Computational fluid dynamics (CFD) simulation plays a very important role by adding cost effectiveness.

KEYWORDS: CFD, Flow Coefficient, Intake Port, Swirl Ratio

INTRODUCTION

Engine intake port geometry controls swirl ratio, volumetric efficiency and combustion characteristics. Swirl generation of different port configurations has been studied to achieve the optimum swirl ratio that is essential for different engine applications and operating conditions, this would lead to optimum combustion is achieved. [13]

An important step towards the development of an internal combustion engines is the optimization of the inflow through intake ports since the charge movement generated by the intake flow considerably influences the quality of mixture and combustion especially in diesel engines. Combustion chamber flow field characteristics at the time of fuel injection and subsequent interactions with fuel sprays and combustion phenomena are dominant parameters for the engine performance and exhaust emissions levels in a direct injection diesel engine. Exact matching of the engine, fuel injection parameters, bowl shape, compression ratio and scavenging characteristics as well as exhaust gas recirculation are important considerations in a new engine design. The in cylinder flow field is largely determined by the swirl and tumble motion during the intake stroke and the squish flow into and out of a piston bowl. Since the nature of the swirling flow in an

engine under operation is extremely difficult to determine, steady flow tests are often used to characterize the swirl and discharge coefficient. So that it becomes important steps in a new engine design to come up with the geometrical shape of an intake port or valve that produces the optimum swirl ratio. [13]

Turbulence inside the cylinder is high during the intake and then decreases as the flow rate slows near the bottom dead centre (BDC). It increases again during the compression stroke as swirl, squish and tumble increase near the top dead centre (TDC). The high turbulence near TDC when ignition occurs is much required for combustion process of the engine. It breaks up and spreads the flame front many times faster than that of a laminar flame. Both fuel and air is consumed in a very short time, whereby self- ignition and knock are avoided. This turbulence is enhanced by the expansion of engine cylinder during the combustion process. [11]

The main function of an air intake system is to supply the engine with clean air with correct amount for the required air to burn in the manifold chamber. The flow efficiency of the intake system has a direct impact on the power the engine is able to deliver. [6]

The main task of an intake port is to distribute air between cylinder properly, identical distribution of air to cylinders is vital for an optimized engine. An uneven air distribution leads to non-uniform cylinder volumetric efficiency, power loss and increased fuel consumption. During the operation of an IC engine, pressure waves occur because of pressure drop in cylinders in intake strokes. [8]

From the review of literature, it can be noted that, design of inlet manifold configuration is very important in an IC engine. Hence, this study looks up on the effect of helical, spiral, and helical-spiral combined configuration on the induced mean swirl velocity in the piston bowl at TDC, swirl ratio and flow coefficient during suction and compression stroke, turbulent kinetic energy variation and volumetric efficiency at various engine speed. [5]

In an Internal Combustion Engine the performance, efficiency and emission formation depends on the formation of air-fuel mixture inside the engine cylinder. The fluid flow dynamics plays an important role for air-fuel mixture preparation to obtain the better engine combustion, performance and efficiency. Due to the extreme conditions inside a typical IC-engine (high combustion temperatures and pressures, precipitation of soot and other combustion products, etc.) experimental techniques are sometimes limited in approaching the above mentioned problem.

Steady State Analysis

The pressure loss coefficients for individual can be determined with using steady state simulation. This information can be obtained from a steady flow test (flow bench) too. The boundary conditions (BC) in steady state simulation are constant pressure. [8]

The main aim of steady state analysis is to find the air flow pattern for different cylinder opening conditions. Further eddies formation during suction stroke can be analyzed. Steady state analysis can provide the loss coefficients but it cannot provide any information about an IM performance in the transient operating conditions. [4]

Unsteady State Analysis

Steady state study can be fast and can provide the loss coefficients but this information cannot provide any information about an IM performance in the operating situation. Unsteady state simulation can predict how an IM work

under real conditions. The boundary conditions are not longer constant but time variant. These boundary conditions are obtained from the 1-D gas dynamics analysis by using the Wave code. [8]

Flow through an intake manifold is dependent on the time since crank angle positions vary with respect to time. Unsteady state simulation can predict how an IM work under real conditions. The boundary conditions are no longer constant but vary with time. [4]

MODELLING PROCEDURE

Steady state air flow calculations are performed for three different intake valve lifts viz. low lift, medium lift and high lift to investigate the flow features. Sufficient mesh refinement has been provided near the throat area because the flow velocity changes rapidly in this region and capturing the gradients is key for an accurate simulation. The calculations are performed by solving compressible Navier-Stokes equation for mass, momentum and energy. Also two equation turbulence model, Realizable $\kappa - \varepsilon$ is used to capture the flows involving rotation, boundary layer under strong adverse pressure gradients, separation and recirculation. Pressure based segregated solver is used to solve the transport equation for mass, momentum and energy. Pressure field is obtained using SIMPLE algorithm for Pressure-Velocity coupling. [12]

The study is expected to explore the potential of using CFD tool for design and optimization of engine inlet manifold. The commercial CFD code STAR-CD is used for the analysis of flow. The CFD package includes user interfaces to input problem parameters and to examine the results. The code contains three elements: - 1. Pre-Processor 2. Solver 3. Post Processor [5]

The CFD code of STAR-CD for finite volume method has been utilised to solve the discretized continuity and Navier-Stokes equations. This CFD code is commonly used for the internal combustion engine and has the high capability of solving the transient, compressible, turbulent-reacting flows with sprays on the finite volume grids with moving boundaries and meshes. Fully hexahedral meshes of intake and exhaust ports and combustion chamber are utilised. The second upwind differencing scheme (MARS) as the spatial discretisation is used for the momentum, energy and turbulence equations. At the beginning and the end of the intake stroke, when the valve lift is quite small, there are high local velocities in the discharge zone. Therefore, the time step must be as small as enough.[11]

MESH GENERATION

The computational domain for the CFD calculation covers the intake ports and valves, the cylinder head and the piston bowl as shown in Figure 3. The number of cells varies from 1,00,200 cells in TDC and around 250,000-450,000 cells in BDC, The fine grid structure is necessary for mesh snapping during the valve movement. The hexahedral cells have been adopted for the mesh generation because they provide a better accuracy and stability compared to the tetrahedral cells. The much important motivation about the use of hexahedral cells is the requirements of moving meshes and boundaries to accomplish the CFD calculation. [11]

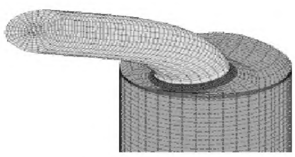

Figure 1: The Computational Domain of the Engine Model

Academic Engine Model

Table 1: Specification of the Engine Model for Computational Condition

Engine Parameters	Value
Number of cylinder	4
Type	Inline
Displacement Volume	1560
Bore	90
Stroke	89
Connecting rod length	135
Crank radius	45
Compression ratio	18
Intake valve opening	14
Intake valve closing	50
Exhaust valve opening	44
Exhaust valve closing	15
Maximum intake valve lift	8.3
Maximum exhaust valve lift	7.8
Combustion chamber	Bowl piston

CALCULATION PARAMETERS

Flow Coefficient: [12]

The flow coefficient ($\alpha\ k$) is defined as the ratio of the actual or measured mass flow rate at standard condition and the theoretical mass flow rate. Cylinder bore diameter is used as characteristic length for calculating the theoretical mass flow rate.

$\alpha_k = m_{std}/m_{theo}$

Swirl Ratio: [12]

Swirl ratio is defined as the ratio of circumferential air speed in the cylinder to the axial speed of the air flow in the cylinder.

Swirl ratio = Circumferential Velocity (C_u)/Axial velocity (C_a)

P_{std} = 100000N/m2

T_{std} =288.7k

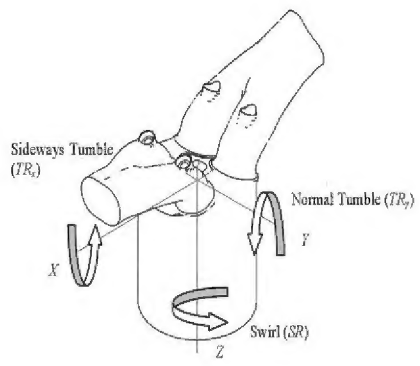

Figure 2: Schematic View for the Definition of Swirl and Tumble Axes and Directions (SR: Swirl Ratio)

RESULTS AND DISCUSSIONS

The table below shows discharge coefficient and swirl ratio obtained from given engine specification of academic model. (Table 2)

Table 2

Lift	Flow Coefficient	Swirl Ratio
Low lift	0.2198	0.920
Medium lift	0.5478	0.724
Maximum lift	0.6547	0.789

The table below shows flow coefficient and swirl ratio obtained from steady state CFD calculation.

Table 3

Lift	Flow Coefficient	Swirl Ratio
Low lift	0.2143	1.264
Medium lift	0.4506	0.911
Maximum lift	0.6203	0.710

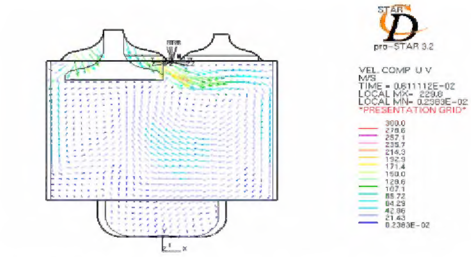

Figure 3: Velocity Vector at Max. Valve Lift Position for Helical-Spiral Manifold

Low lift Medium lift High lift

Figure 4: Pressure Countor

ACKNOWLEDGEMENTS

I would like to thank Prof. Atul A. Patil and Prof. V.H.Patil HOD of Godavari college of Engineering, Jalgaon providing valuable inputs on Optimization for Intake port

CONCLUSIONS

We as budding engineers tend to mixed up technical terms in "Optimization for Intake Port". The main task of an intake port is to distribute air between cylinder properly, identical distribution of air to cylinders is vital for an optimized engine.

By using the academic engine specification model here the results were compared between flow coefficient and swirl ratio obtained from steady state CFD calculation. The calculation was carried out at discrete valve lifts to obtain air

flow rate and rotational speed. CFD and experimental results were compared for its accuracy both flow coefficient and swirl ratio calculated through CFD which gives satisfactory results.

We have only gives you an overview of the fascinating knowledge of intake port optimization. It is regretted, however, that despite careful scrutiny of the proofs several spelling mistake have nevertheless remained in the paper.

REFERENCES

1. Abhilash M Bharadwaj. K Madhu, Seemanthini J., Vismay K.G., Aravind T & Anand M Shivapujii[1], International Journal on Theoretical and Applied Rsearch in Mechanical Engineering (IJTARME), ISSN:2319-3182, Volume-1, Issue-2, 2012, Study of swirl and Tumble Motion using CFD, Department of Mechanical Engineering, Atria Institute of Technology, Bangalore[1], Research Scholar, Indian Institute of Science, Bangalore.
2. Giuseppe Tesolin-University of Trieste, Aiders: Peter Tibaut- AVL List GmbH, Paolo Geremia- ES.TEC.O s.r.l., Francesco Pinto- University of Trieste, TCN CAE 2005, Coupled modeFRONTIER/FIRE Approach for I-C Engine Intake Port Optimization.
3. N. Trigui1, V. Griaznov1, H. Affes1 and D. Smith1, Oil & Gas Science and Technology – Rev. IFP, Vol. 54 (1999), No. 2, pp. 297-307, CFD Based Shape Optimization of IC Engine, 1 Ford Motor Company, 20000 Rotunda Drive MI 48121-2053, Dearborn - United States.
4. S. Karthikeyan, R. Hariganesh, M.Sathyanadan, S. Krishnan, P. Vadivel, D.Vamsidhar, International Journal of Engineering Science and Technology (IJEST), ISSN: 0975-5462, Vol. 3 No. 4 Mar 2011. Engine Product Development Department (Sunrise), Ashokleyland Technical Center Chennai, Tamil Nadu, India and CFD Department, Defiance Technologies Chennai, Tamil Nadu, India.
5. Benny Paul1*, V. Ganesan2, International Journal of Engineering, Science and Technology, Vol. 2, No. 1, 2010, pp. 80-91, Flow field development in a direct injection diesel engine with different manifolds, 1,2 Internal Combustion Engines Laboratory, Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai 600 036, India.
6. D.Ramasamy, Zamri.M, S. Mahendran, S.Vijayan, Design Optimization of Air Intake System (AIS) of 1.6L Engine by Adding Guide Vane , ISBN:-978-988-18210-4-1, ISSN:-2078-0958 (print): , ISSN:2078-0966 (online), Proceeding of the International Multiconference of Engineers and computer scientists 2010 Vol-II IMECS 2010, March 17-19,2010, Hong Kong.
7. Padmesh Mandloi, Gunjan Verma, Nafems World Congress 2009 June 16th-19th 2009 Crete, Greece, Design Optimization Of An In- Cylinder Engine Intake Port, ANSYS Fluent India Pvt. Ltd., India Arnaud Boland, ANSYS, Belgium.
8. Negin Maftouni , Reza Ebrahimi, Proceedings of the 3rd BSME-ASME International Conference on Thermal Engineering 20-22 December, 2006, Dhaka, Bangladesh., Intake Manifold Optimization By Using 3-D Cfd Analysis With Observing The Effect Of Length Of Runners On Volumetric Efficiency, K. N. Toosi University of Technology, Tehran, Iran.

9. He Changming*, Xu Sichuan**, Zuo Chaofeng***, Chen Xin****, Li Chuanyou***** ISSN 1392 - 1207. MECHANIK 2011. 17(6) 643-648, Multi-valve intake port parametric design and performance optimization of the horizontal diesel engin *Tongji University, Shanghai, China, **Tongji University, Shanghai, China, ***Goptima, Shanghai, China, ****Goptima, Shanghai, China, *****Goptima, Shanghai, China.

10. Jorge MARTINS, Senhorinha TEIXEIRA, Stijn Coene, Proceedings Of COBEM 2009, 20th International Congress Of Mechanical Engineering. “Design of an Inlet Track of a Small I. C. Engine for Swirl Enhancement copyright” © 2009 By ABCM November 15-20, 2009, Gramado, RS, Brazil.

11. Wendy Hardyono Kurniawan, Shahrir Abdullah, Kamaruzzaman Sopian, Zulkifli Mohd. Nopiah and Azhari Shamsudeen, Journal - The Institution of Engineers, Malaysia (Vol. 68,69, No.1,March- 2008) CFD Investigation of FluidFlow and Turbulence Field Characteristics in a Four-Stroke Automotive Direct Injection Engine, Department of Mechanical and Materials Engineering, Faculty of Engineering Universiti Kebangsaan Malaysia.

12. Scott Morton, Mercury Marine, Paul Radavich, Mercury Marine, Vinodh kumar B, Sivagaminathan N, Gopalakrishnan N Larsen And Toubro Limited, IES, **“**Air Flow And Charge Motion Study of Engine Intake Port”.

13. Javad Kheyrollahi, Mojtaba Keshavarz, DESA, Iran, CIMAC Congress 2010, Bergen PAPER NO.: 306 “Optimization of Intake port shape in a DI Diesel Engine Using CFD Flow Simulation”.